



Modelling of large displacements and large strains in coupled electromagnetic/solid mechanics/heat transfer problems

J.R. Alves, J Barlier, S Marie, C Beraudo, F Bay

► To cite this version:

J.R. Alves, J Barlier, S Marie, C Beraudo, F Bay. Modelling of large displacements and large strains in coupled electromagnetic/solid mechanics/heat transfer problems. 8th International Conference on Electromagnetic Processing of Materials, Oct 2015, Cannes, France. hal-01334238

HAL Id: hal-01334238

<https://hal.science/hal-01334238>

Submitted on 20 Jun 2016

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Modelling of large displacements and large strains in coupled electromagnetic/solid mechanics/heat transfer problems

J.R. Alves^{1a}, J. Barlier^{2a}, S. Marie^{2b}, C. Beraudo^{2c}, F. Bay^{1b}

¹ MINES ParisTech, PSL – Research University, CEMEF - Center for Material Forming – CNRS UMR 7635 CS 10207, F-06904 Sophia-Antipolis Cedex, France

² Transvalor SA, Parc de Haute Technologie, 694 av du Dr Maurice Donat, 06255 Mougins, France

Corresponding author: jose-rodolfo.alves_zapata@mines-paristech.fr

Abstract

The current work deals with the modelling strategies for the improvement of coupled electromagnetic thermo-mechanical problems due to induced large deformations or displacements. We introduce a sequentially-coupled algorithm that includes two types of mesh management models. The methodology enables automatic switching between the mesh management models in order to balance computational time and results accuracy.

Key words : Electromagnetic Forming, Induction Heating, Finite Elements, Mesh adaptation.

Introduction

Several EPM processes involve large displacements or large strains of the solid bodies. We can mention magnetic forming applications [1] – where the part to be formed undergoes large strains and strain rates, – induction heat treatment applications where the inductors may have a large motion with respect to the part to be heated/treated. This problem is one of the core subjects shown in this work.

We shall present here the work carried out on the coupled simulation package MATELEC/FORGE® [2], where MATELEC is the FORGE® electromagnetic module developed in our laboratory. This tool enables solving the electromagnetic equations by means of the finite elements method. The sequentially-coupled approach between MATELEC and the thermo-mechanical solver enables an optimized treatment of the physics, preserving reasonable computational cost for complex cases. The main industrial applications treated here are electromagnetic forming and induction heating.

A full finite elements approach has been retained for solving the electromagnetic equations. This means that besides the solid parts, a surrounding medium (air) has to be meshed in order to allow electromagnetic coupling between the conductors. The movement of the solids within the surrounding domain is included in the solver by means of automatic r-adaptation or remeshing techniques.

Simulation Strategy

One of the main challenges for solving multi-physical problems is how to set-up the resolution scheme in terms of the coupling between the physics. As shown in Fig. 1, all the physical aspects are highly dependent one on each other. The two main options that can be proposed are: (a) To include all the physics in a single solver for a simultaneous resolution (strongly-coupled). Or (b) solve each physics in a separate fashion, communicate and correct iteratively. We have chosen to follow this sequentially-coupled approach because solving the physics separately has the advantage of allowing independent management of the meshes – meaning that each problem will have a finite element space adapted to its own requirements.

Fig. 2 describes the global work flow to accomplish a single time step increment. The coupling between the solvers is driven by the communication of: The Lorentz volumetric forces (\vec{F}) and the Joule heating term (\dot{Q}) at each element of the solid body from the EM module. The temperature evolution (T) and the velocity field (\vec{v}) or the new position (\vec{x}) of the nodes of the solid body are transmitted from the TM module. The main difference between the electromagnetic forming application and the induction heating process is that the communication is done after each time-step increment in the first case or after a full electric period for the latter.

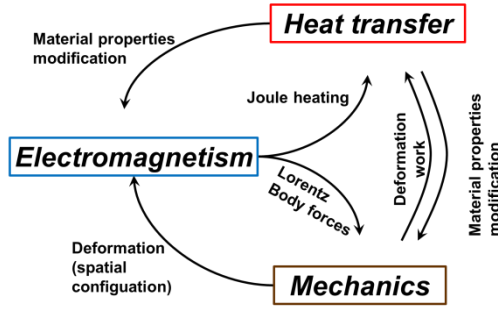


Fig. 1: Scheme of the physics coupling and interaction

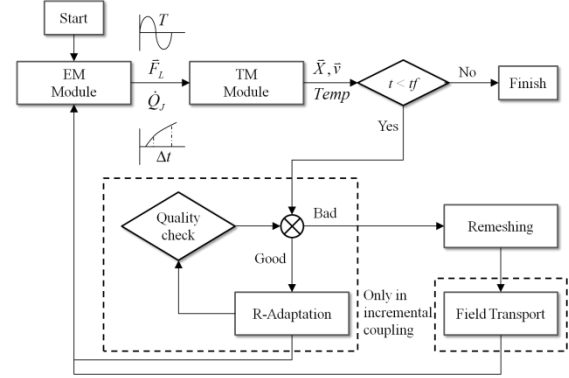


Fig. 2: Global algorithm of the coupling between the solvers and the mesh adaptation modules.

The electromagnetic wave propagation phenomenon is described by the Maxwell's equations. The latter is commonly expressed in potential form as shown in set (1) for many numerical applications.

$$\sigma \frac{\partial \vec{A}}{\partial t} + \frac{1}{\mu_0} \nabla \times \frac{1}{\mu_r} \nabla \times \vec{A} = -\sigma \nabla \phi \quad (1.a)$$

$$\nabla \cdot (\sigma \nabla \phi) = 0 \quad (1.b)$$

Where, \vec{A} is the magnetic potential, ϕ is the electric potential, σ is the material electric conductivity, μ the magnetic permeability, ϵ the electrical permittivity, subscript "0" represents the value for void and "r" the relative value with respect to the void.

The thermo-mechanical problem is described by the conservation laws. Equation (2) represents the conservation of energy with heat sources (induction heating) and the effect of self-heating by mechanical deformation. C_p denotes the heat capacity, the main unknown T the temperature, and k accounts for the heat conduction within the body.

$$\rho C_p \frac{dT}{dt} - \nabla \cdot (k \nabla T) = \dot{q}_{vol} + \underline{\underline{\sigma}} : \underline{\underline{\dot{\epsilon}}} \quad (2)$$

Equations (3) represent the conservation of linear momentum (3.a) and the conservation of mass for a compressible body (3.b).

$$\rho \frac{d\vec{v}}{dt} - \nabla \cdot \underline{\underline{\sigma}} = \vec{b} \quad (3.a)$$

$$\nabla \cdot \vec{v} = -\frac{\dot{P}}{\kappa} \quad (3.b)$$

Where $\underline{\underline{\sigma}}$ is the Cauchy stress tensor, \vec{b} represent any volumetric force such as gravity (which is not taken into account in our analysis) or the Lorentz body forces, in which case

$$\vec{b} = \vec{F}_{lorentz} = \vec{J} \times \vec{B} \quad (4)$$

The inertia is described by the material density ρ and the material time derivative of the velocity $d_t \vec{v}$. The hydrostatic pressure is represented by P and the bulk modulus κ defines the compressibility in the elastic deformation regime.

Parallelism

3D simulation of coupled electromagnetic/solid mechanics/heat transfer problems with large displacements and large strains leads to wide number of increments and can be CPU time intensive. Therefore use of parallel clusters machine is strongly recommended for the resolution of such long processes.

The coupled MATELEC/FORGE® simulation strategy makes it easy to handle two levels of parallelism: the parallelization of the FORGE® thermo-mechanical Finite Element software which is already efficiently done [3] and the parallelization of the electromagnetic algorithm itself [4]. In both cases the parallelization method is similar, based

on a SPMD (Single Program on Multiple Data) strategy where each processor runs a full version of the code on a partitioned mesh. Within FORGE® the SPMD paradigm is based on the tetrahedral elements partitioning of the mesh billet because the thermo-mechanical computation is carried out only on the work piece.

The electromagnetic formulation consists in discretizing in space of the whole domain in one single mesh (parts, inductors, air) using Nedelec edge elements [5]. Thus the magnetic vector potential/electric scalar formulation described in the previous paragraph leads finally to resolve symmetric linear systems with advantageous small bandwidth. It ensures a good performance on parallel linear iterative solvers which are solved via the external PETSc library [6]. The parallel Finite Element assembly is based on tetrahedral elements partitioning of the whole mesh domain. A nodes partitioning data structure is built with a specific renumbering where all the nodes of a parallel sub-domain are on the same processor. It enables to minimize communications between processors when solving linear systems. A parallel data structure partitioning by elements, nodes and edges is implemented to update and synchronize global vectors and scalars between parallel sub-domains.

The edges formulation of the magnetic potential induces a large number of degrees of freedom in comparison to the parallel communication inter processors which is proportional to the number of nodes of the subdomains interfaces. Therefore, the parallelization strategy provides a good parallel efficiency on a large number of cores as well on entry level parallel computers.

Models

The first model consists in a magnetic pulse welding simulation of an aluminum tube with a copper cylinder as shown in Fig. 3. In the EM module we take into account the inductor, the work piece and a surrounding air box in order to allow the propagation of the electromagnetic waves. Besides the EM computation, we also define a coupling with a RLC circuit model in which the electromagnetic forming machine parameters are defined. The electric current evolution is then computed as a result from the coupling between the circuit and the FE model. In the thermo-mechanical module the work piece is defined again together with any tools/holders that may be needed for contact conditions. Since the meshes are in principle different, the communication is carried out by interpolation of the data at the integration points.

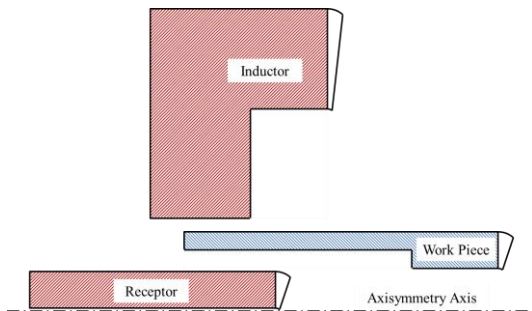


Fig. 3: Magnetic pulse welding of an aluminum A2024 tube with a copper cylinder.

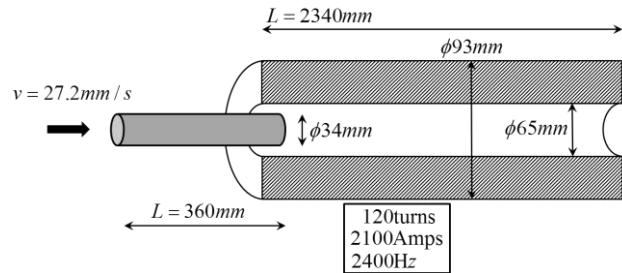


Fig. 4: Induction heating case of an steel A304L billet.

The second model consists on an induction heating process of a steel bar for heat treating the outer surface by means of an AC current of 2.1kAmps as shown in Fig. 4. For this process, the EM module computes a complete period of the AC signal and the Joule heating transmitted to FORGE® corresponds to the rms value on the period. Then, FORGE® computes the evolution of the temperature field until a defined variation before the coupling is reestablished.

Results

For the first case corresponding to the magnetic pulse welding application, we had small displacements of about 1.5 mm but large deformations on the work part reaching 33% before impact. This condition allows taking full advantage of the algorithm presented Fig. 2. The small incremental displacements enable to use r-adaptation algorithm to follow the geometry evolution as can be seen in Fig. 5. At this stage, several elements have in surrounding air have withstood critical deformation. This condition automatically launches the remeshing module at the next electromagnetic time step increment.

The thermo-mechanical model also counts with its automatic remeshing/mesh adaptation module. While several options are available, we show in this study how it is possible to setup a “box” region near the impact zone. This is useful to create a fine mesh zone just prior the contact condition as can be seen in Fig. 6.

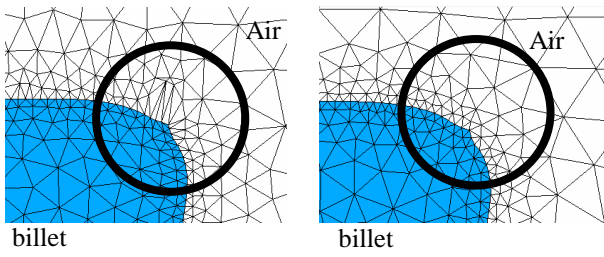


Fig. 5: Mesh adaptation of the air around the deformed billet. (Left) R-adaptation until $t=3.2\mu s$. (Right) Automatic mesh generation at $t=3.4\mu s$.

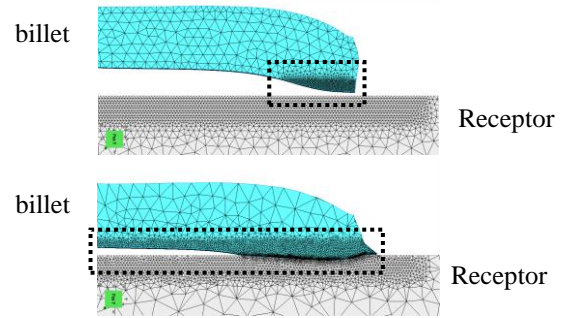


Fig. 6: Mesh refinement in the thermo-mechanical computation to capture impact conditions. (Up) Before impact. (Bottom) After/during impact.

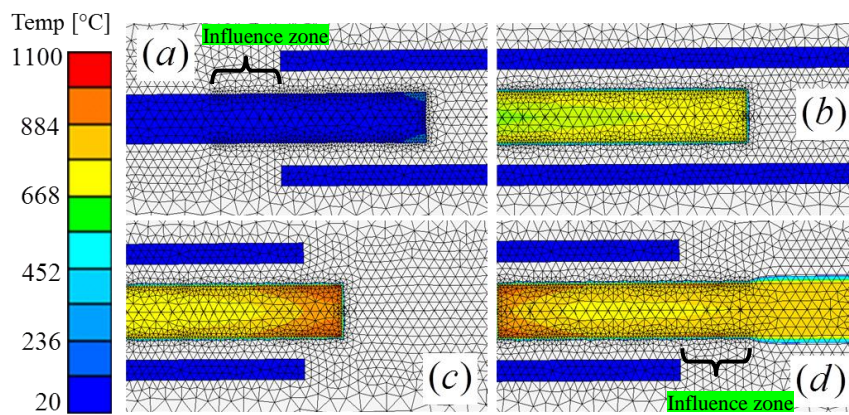


Fig. 7: Mesh adaptation around the heated bar in order to allow the displacement through the inductors center.

While in EMF large deformation with small to medium displacements are expected, in induction heating application we will most likely face large displacements (of the inductor and/or the treating piece) with small to negligible deformations. In this case, the r-adaptation algorithm is not adapted to capture the movement. In such case, only the remeshing module is used. Using a predefined parameter, an influence zone is defined from the inductor. Within this zone, the material parameters are used to automatically define the mesh size in order to respect the skin depth effect at the surface of the work piece as is shown in Fig. 7.

Conclusions

One of the complexities associated to multi-physical simulation of electromagnetic forming or induction heating processes is the careful management of the mesh generation and the continuous tracking of displacements and deformations. In this work we have presented the algorithmic procedure that was defined within the FORGE®/MATELEC interface in order to automatically generate and adapt the global electromagnetic mesh due to the displacement and/or deformation of the immersed solids.

References

- [1] V. Psyk, D. Risch, B. L. Kinsey, A. E. Tekkaya, and M. Kleiner (2011), Electromagnetic forming -A review, *Journal of Materials Processing Technology*, vol. 211, no. 5, 787–829.
- [2] C. Fourment, J. Barlier, M. Barbelet, P. Lasne, and D. Cardinaux (2014), State-of-the-Art 3D simulation Tool as a Strategic Investment, *SAE Technical Paper*.
- [3] T. Coupeze and S. Marie (1996), Parallel simulation of 3D forming processes, in *Proceedings of the International EUROSIM Conference*, 221–228.
- [4] P. De Micheli, A. Settefrati, S. Marie, J. Barlier, P. Lasne, B. Scholtes, M. Bernacki, and F. Bay (2015), Towards the simulation of the whole manufacturing chain processes with FORGE®, in *New Developments in Forging Technology 2015*.
- [5] J. C. Nédélec (1986), A New Family of Mixed Finite Elements in R 3, *Numer. Math*, vol. 50, 57–81.
- [6] S. Balay, K. Buschelman, V. Eijkhout, W. Gropp, D. Kaushik, M. Knepley, L. C. McInnes, B. Smith, and H. Zhang (2012), *PETSc Users Manual*, Tech. Rep. Argonne National Laboratory.